DTU Library

Forensic finite element study of skull fracture

Mikkelsen, Lars Pilgaard; Sölvadóttir, A.E.; Thomsen, H.R.; Lynnerup, N.

Publication date:
2009

Forensic finite element study of skull fracture

L.P. Mikkelsen[1), A.E. Sölvadóttir[2)], H. R. Thomsen[2)] and N. Lynnerup[3)]

[1)] Materials Research Division, Risø DTU, Technical University of Denmark, DK-4000 Roskilde, Denmark

[2)] Novo Nordisk A/S, Hillerød, Denmark.

[3)] Department of Forensic Medicine, University of Copenhagen, Copenhagen, Denmark.

An important issue in forensic medicine is to determine how a specific skull fracture arose. This information can be critical for establishing if a criminal offense has taken place. The medical examiner chiefly relies on his experience, since only a few scientific methods are considered sufficiently reliable. At the Institute of Forensic Medicine, University of Copenhagen, all autopsy cases are CT-scanned, but the obtained CT-scan data is mainly used as a visualization and documentation tool. During the last decade computers and finite element codes have reached a stage where accurate numerical finite element predictions of complex fracture processes in large 3-D structures are possible, e.g. a full 3-D finite element simulation of a skull fracture. In addition, tools are available transfer CT-scan data to a 3-D finite element model. It may thus be possible to utilize CT-data in a more pro-active way in forensic medicine.

Here, we demonstrate such a procedure using the software Amira on a CT-scan from a forensic case. A 3-D finite element model of the skull was extracted, and as a first step the skull material was considered a homogeneous isotropic linear elastic material. Even though only a rather simple geometrical part of the skull was considered, some manual work improving the achieved mesh was necessary in order to get a workable model. In the subsequent finite element simulation, a simple element-removing algorithm describing the fracture process was used, and the skull was simulated using a commercial dynamic finite element code (Abaqus/Explicit). The skull was exposed to a short dynamic impact from a blunt object. Based on the simulations, a large sensitivity of the fracture modes was found even for small changes in the initial speed of the incoming object.

The modeled fracture patterns differ somehow from the actual fracture pattern of the specific case. This may be due to some of the simplifications used in the model. A more realistic simulation should take into account a more realistic cranial vault structure, incorporating a spongy bone layer with a low density between two compact bone layers. In addition, the effects of the sutures between the single cranial bones should be included, as well as a more realistic fracture process simulation. Regarding the fracture process, possibilities could be to use the new “eXtended Finite Element Method” (XFEM) feature in the static Abaqus/Standard or the “Virtual Crack Closure Technique” (VCCT) implemented in the dynamic Abaqus/Explicit. An Abaqus/Standard simulation could be relevant, as the time scale for the fracture process in the present simulations is found to be an order of magnitude larger than the stress wave propagation.